STUDENT: Have you ever wondered how rivers flow, or how fluid travels inside pipes? Watch this video and you will learn how. If you apply Newton's second law to fluid motion, and at the same time, consider the viscosity of the fluid, you will get the equations you need to describe fluid flow. The equations that you get this way are called the Navier-Stokes equations. This is the general form of the Navier-Stokes equations.

Solving this equation in this form is not easy. Interestingly, mathematicians have not yet proven that solutions exist to the Navier-Stokes equations in three dimensions. To make our life easy, let's consider motion for fluid in one dimension. The Navier-Stokes equations then become this. As you can see, I've broken down the previous problem in three dimensions into just one dimension, the x dimension.

The main body force that is Fx acting on a fluid will be the force due to gravity. If the fluid is flowing at an angle, theta, to the horizontal, then the component of gravity in the direction of fluid motion is going to be g sine theta.

We now have an equation that looks much easier than a general form. As you can see, this is just a restatement of Newton's second law, with the viscosity term included. On the left side of this equation, you get the net force, acting on a unit volume of the fluid. The three terms on the right side of the equation, which are governed by the viscosity, the pressure gradient, and the force due to gravity together, give the net force acting on a unit volume.

As you can see from this term here, this is a second order differential equation. To solve this, we will have to impose two boundary conditions. For the river problem, we will consider the velocity at the bottom of the river to be 0. This is true because the river bed imposes resistance to the fluid layer just above it, preventing its motion. The velocity of the river reaches a maximum towards the top of the river. So we get the condition that dVx over dy equals 0 at the top.

For the pipe problem, using the same argument, the velocity at the rim of the pipe is 0. And at the center of the pipe, dVx over dr equals 0, where r in this case is the distance from the center of the pipe to the point we are looking at.

Now that we have our differential equation in one dimension and the two boundary conditions needed to solve it, there is nothing left to do but to solve. We can do this using Mathematica. I was able to solve this equation using Mathematica.

In the pipe, the velocity at a distance, r, from the center is given by this formula. And in the river, the velocity the height, y, in the riverbed is given by this formula. If you want to know the average velocity inside the pipe or inside the river, all you have to do is add up each individual velocity at all possible radii, or heights, you can think of. And then divide that sum by the area that you added it over. It's just integrating and dividing.

Mathematica gave me this result. You can see that the average velocity of the fluid in a pipe and in a river increases if you increase the density of the fluid, or if you increase the inclination of the pipe or the slope of the river to the horizontal.

Also, if you increase the negative pressure gradient, the velocity increases. A lower viscosity of the fluid will increases the average velocity. This is all intuitive, but wouldn't it be cool if we could actually model how the velocity profile looks like inside a pipe and inside a river? I'll do this using Mathematica.

I will start with the river problem. First, I will create a grid of coordinates. For each coordinate on this grid, I will assign a vector. This vector will correspond to the velocity at that point. We know how this velocity looks like because we solved the differential equation for velocity. Once we have vectors for each of these points, you'll get a picture that looks like this.

As you can see, at the bottom of the river, where y equals 0, you have 0 velocity. As you increase y, the velocity increases. Let's look at just one slice of the previous image. We'll see what happens when you change parameters. When you change mu, the average velocity decreases. Think of these dark blue dots as fixed. These are the reference frame. With higher mu, you have lower velocities. You can do the same for other parameters.

I used a similar method to model the velocity profile inside a pipe. I first created a circular grid, and then I assigned each point on the grid a vector. As you can see, on the rim of the pipe, you have 0 velocity. And towards the center, this dark blue line here, you have the highest velocity.

[MUSIC PLAYING]

I hope you now understand how fluids flow in pipes and rivers, and that you will be able to apply Navier-Stokes equations to solve similar problems. Thank you for watching.

[MUSIC PLAYING]